

## Introduction to Autodesk Inventor Nastran

Autodesk Inventor Nastran is a powerful Finite Element Analysis (FEA) package built into Inventor. This course teaches the capabilities of Nastran including complex stress, impact, fatigue, vibration and thermal analysis.

### Course Modules

- *Introduction to Finite Element Analysis*
- *Idealisations*
- *Contacts*
- *Constraints*
- *Leads*
- *Meshes*
- *Reviewing Results*
- *Sections*
- *Animations*

### Study Types

- *Linear Static Analysis*
- *Nonlinear Static Analysis*
- *Normal Modes Analysis*
- *Linear Buckling Analysis*
- *Nonlinear Buckling Analysis*
- *Prestress Static Analysis*
- *Prestress Normal Modes*
- *Direct Transient Response*
- *Modal Transient Response*
- *Impact Analysis*
- *Nonlinear Transient Response*
- *Direct Frequency Response*
- *Modal Frequency Response*
- *Random Frequency Response*
- *Shock/Response Spectrum*

### Prerequisites

Prior knowledge of Inventor Solid Modelling is required and Finite Element Analysis experience is beneficial but not mandatory.

### Course Duration

2 days

### Next Steps

A Bespoke Course